Winter 2011 Semester **3 Apr 2011**

Wayne State University College of Engineering

ME 5620 Fracture Mechanics in Engineering Design

Case Study Project

Prepared by:

Gerald Byrd

Instructor: Dr. Nripen Saha

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE **03 APR 2011**	2. REPORT TYPE **N/A**	3. DATES COVERED **-**
4. TITLE AND SUBTITLE **Case Study Report on Track Pin**		5a. CONTRACT NUMBER 5b. GRANT NUMBER 5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) **Gerald Byrd**		5d. PROJECT NUMBER 5e. TASK NUMBER 5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **US Army RDECOM-TARDEC 6501 E 11 Mile Rd Warren, MI 48397-5000, USA**		8. PERFORMING ORGANIZATION REPORT NUMBER **21686**
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) **US Army RDECOM-TARDEC 6501 E 11 Mile Rd Warren, MI 48397-5000, USA**		10. SPONSOR/MONITOR’S ACRONYM(S) **TACOM/TARDEC/RDECOM** 11. SPONSOR/MONITOR’S REPORT NUMBER(S) **21686**

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release, distribution unlimited

13. SUPPLEMENTARY NOTES
Submitted to Wayne State University College of Engineering as a Case Study Project for ME 5620 Fracture Mechanics In Engineering Design, The original document contains color images.

14. ABSTRACT

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**	**SAR**	**14**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

Executive Summary

The objective of the project was to analyze a test failure on a track pin that did not meet the durability requirement. The case study consists of a fracture example subjected to fatigue loading. Several track pins consisted of the same fracture failure which was located at the middle of the pin. The customer defined the crack initiation as the failure criteria; therefore crack propagation or crack growth analysis was excluded from the analysis.

The development of the track pin consisted of three different designs. The first design had a thin-wall section and surface treatment *A*. The second design was modified to a thick wall section with a surface treatment *B*. The third design went back to the original thin-wall section, but modified its original surface treatment to surface treatment *C*. A FEA and Fatigue analysis was conducted to determine if the new designs meet the durability requirement of 5000 miles.

Abaqus and HBM nCode DesignLife was the FEA and Fatigue/Fracture Software used in the analysis, respectively. There were six loads applied at three locations of the track pin. A unit load was applied in two directions at the two end connectors and the middle of the pin. A durability schedule was created in nCode DesignLife that reference the dynamic load time history data given from a dynamics software to the required 5000 mile durability course breakdown.. The 5000 miles consisted of 15% for Road #1 course, 50% for Road #2 course, and 35% for Road #3 course. The Stress-life (S-N) curves for the three designs were obtained from the material testing group. Goodman method was the parameter used in the Fatigue/Fracture solver to calculate the damage and life cycles.

The fatigue life was predicted for the three designs using the FEA, durability duty cycle, and given material S-N curves. The original design resulted in a fatigue/fracture life prediction of 1940 miles. The crack initiation failure occurred first at the middle of the track pin. This location and the life prediction were validated close to the actual test data. The second and third designs resulted in a prediction of 27700 miles and 11800 miles, respectively. Both designs met the durability requirement of 5000 miles based on the analysis, but Design 3 was selected based on lower mass. It was recommended that additional design iterations would be necessary to minimize mass while satisfying the threshold durability duty cycle of 5000 miles.

Table of Contents

1. Introduction

The project consists of a track pin that is subjected to fatigue loading. This analysis presents a fracture example, which cracks occur at the middle of the track pin due to fatigue at a low number of cycles under a durability requirement. A new design with different surface treatments was introduced to increase fatigue life and also minimize mass.

The track pin is constrained at the end connectors and the middle between the two track bodies. There are several loading conditions applied to the track pin. The loading conditions are defined in detail in the analysis section of the report.

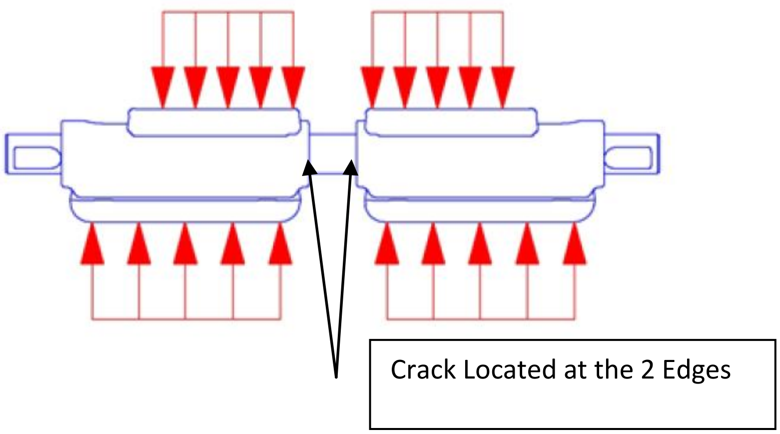

Figure 1: Track Pin Problem

The track pin undergone 3 design changes that consisted of different surface treatments and thinner pins. Table 1 shows the design iterations that were analyzed. It is important to note that Design 1 was the original design and from testing, which failed at an average of 1845 miles.

Table 1: Design Iterations

Design	Thick/Thin Wall Pin	Surface Treatment	Avg. Failure Occurs (miles)
1	Thin	A	1845
2	Thick	B	?
3	Thin	C	?

All the cracks occurred through the entire pin, cutting the pin in half. There is a very short time in the crack propagation after crack initiation. Therefore, it is important to analyze the pin for fatigue which will predict the number of cycles before the crack initiates.

The goal of this problem is to analyze the original pin and new designs, and predict the fatigue life (number of cycles before fracture). The results of this analysis will show whether the new designs results in higher fatigue life, and if it meets the durability requirement. This report will discuss some background of FEM and stress concentration, types of mesh shapes and their corresponding results, and the analysis results of the track pin.

1.1 Finite Element Methods

The Finite Element Method (FEM) is a numerical method for solving problems of engineering and mathematical physics [1]. The solution for structural problems is typically determining the displacements at each node and the stresses within each element making up the structure under specified loading and boundary conditions. The general steps to FEM include: select the element types, select a displacement function, define the strain/displacement and stress/strain relationships, and assemble the element equations to obtain the global equation with boundary conditions, solve for the element strains and stresses, and interpret the results [1]. For commercial codes, most of these steps are done in the background when the commercial code is simulating the model. FEM approach is an easy way to model irregular shaped bodies, especially to those composed of several different materials because the element equations are evaluated individually. Other advantages to FEM include the ability to handle unlimited numbers and kinds of boundary conditions, altering the model relatively easily and cheaply, and handling nonlinear behavior existing with large deformations and nonlinear materials [1].

1.2 Element Quality Criteria and Element Shapes

Element Quality Criteria and element shapes are significant characteristics in FEA modeling, and could influence the accuracy of the results. Table 2 shows the some of the quality criteria for 2-D triangular and quadrilateral meshes used commonly in industry.

Table 2: 2-D Element Quality Criteria

Quality Check		Threshold	Ideal Values
Warpage		> 5	0
Aspect Ratio		> 5	1
Skew		> 60	0
Jacobian		< 0.7	1
Min Angle	Quads	< 45	90
	Trias	< 20	90
Max Angle	Quads	> 135	90
	Trias	> 120	90

The Warpage is the amount by which an element deviates from being planar. The quadrilateral element is divided into two triangular elements along its diagonal, and the angle between the triangular elements' normals is measured as shown in Figure 2. The Aspect Ratio is the ratio of the longest dimension to the shortest dimension of a quadrilateral element [1]. Aspect ratios should rarely exceed 5:1 [2]. As shown in Figure 3, the Skew is calculated by finding the minimum angle between the vector from each node to the opposing mid-side, and the vector between the two adjacent mid-sides at each node of the element as shown in Figure 3. The minimum angle found is subtracted from ninety degrees and reported as the element's skew. The Jacobian measures the deviation of an element from its ideal shape. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. In general, an element yields best results if its shape is compact and regular [1]. For example, the element is poor if the aspect ratio of an element is greater than 5, and the results could vary depending on the number of 'poor' elements in the model.

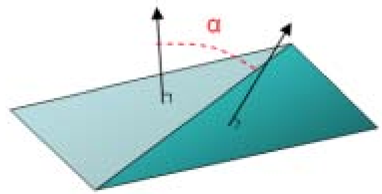

Figure 2: Warpage of Quad Element [3]

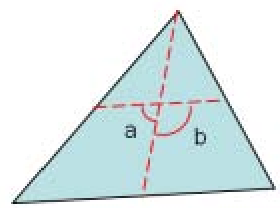

Figure 3: Skew of Triangular Element [3]

It is also recommended using less Constraint Strain Triangular Elements (CST) when generating a mesh which was learned in the last class exercise. As stated in most FEM books, the mesh of CST elements will produce a model that is stiffer than the actual problem [1]. For a bending problem, the element is predicting constant stress within each element, whereas the stress actually varies linearly through the depth of the beam. CST can have false shear stress in places that should not have any shear stress or strain. The shear strain at the incorrect elements absorbs energy and some of the energy is lost in bending. As a result, the CST element is too stiff in bending and results in deformation smaller than the actual solution. This phenomenon is sometimes described as shear locking or parasitic shear [1]. A refinement of elements in the model or decreasing the number of CST elements would help avoid the phenomenon.

1.3 Stress Concentration Factor

Stress concentrations are localized high stress points with an element. Understanding where the stress concentrations are, is important in determining where an object will fracture. Cracks can

begin at stress concentrations. The stress concentration factor is often used to determine the optimal geometry of an object. It can be determined using equation (3).

$$K = \frac{\sigma_{\max}}{\sigma_{nom}} \tag{Eq.1}$$

where σ_{max} is the stress at the re-entrant corner and σ_{nom} is the nominal, uniform, stress. The value of the stress concentration determines how many times larger the maximum stress is over the nominal stress. It is desired to have a small stress concentration factor.

2. Model Setup

The track pin was modeled in the FEA preprocessing tool Altair Hyperworks Hypermesh. Abaqus was the FEA solver used in the analysis. Figure 4 shows the overall representation of the meshed trackpin/bushing. The trackpin is composed of solid 'hex' mesh elements, and the bushings were modeled as springs '1D' mesh elements. Constraints were placed at each free end spring. In this case, the remainder of the track parts would constrain the trackpin/bushing components.

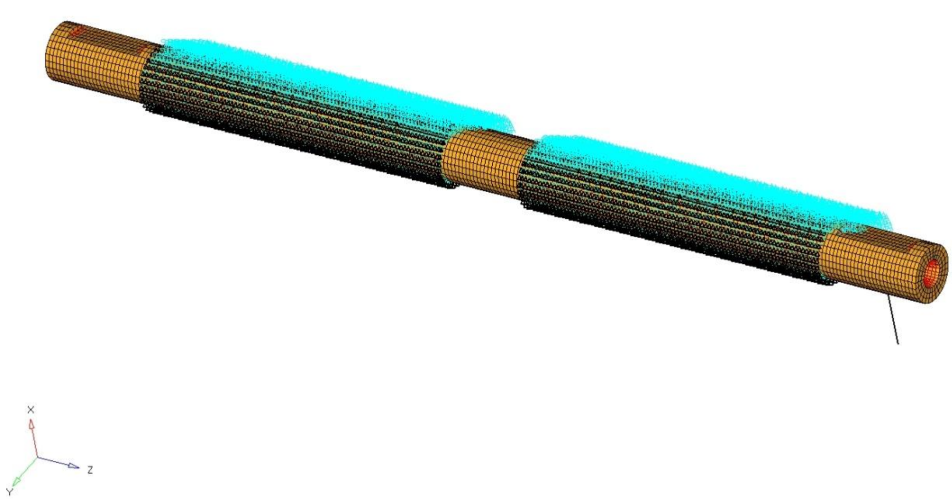

Figure 4: Abaqus Model

Figure 5 is a closer look at the outboard end connector region.

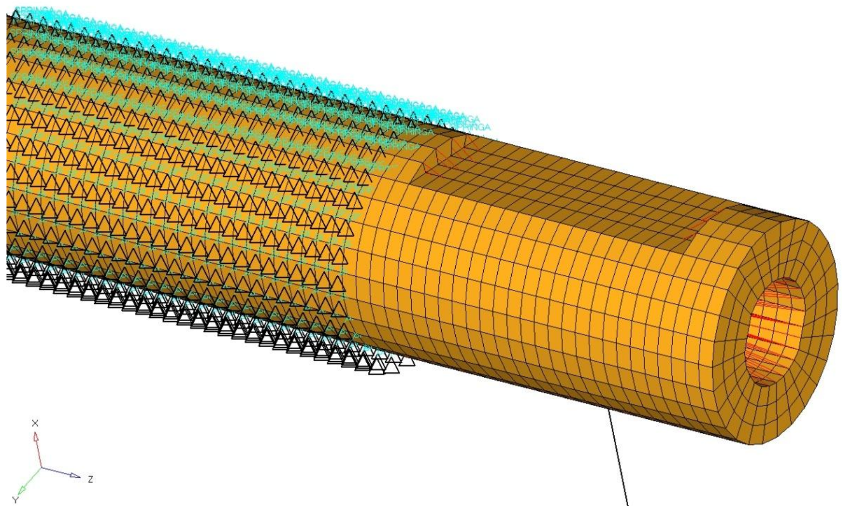

Figure 5: 1st Abaqus Model Close-Up

The point load is distributed across the elements that are positioned inside the end connector. Dynamic software will provide (x & y) direction forces using course. This load case goes the same for the center and the opposite end connector (Total of 3 unit load locations, 2 directions, 6 total loads). A 1N unit load was applied and then superimposed with the other 5 unit loads. The six - 1N loads will be scaled according to the dynamic data. Figure 6 shows the layout of the fatigue life process in nCode DesignLife software.

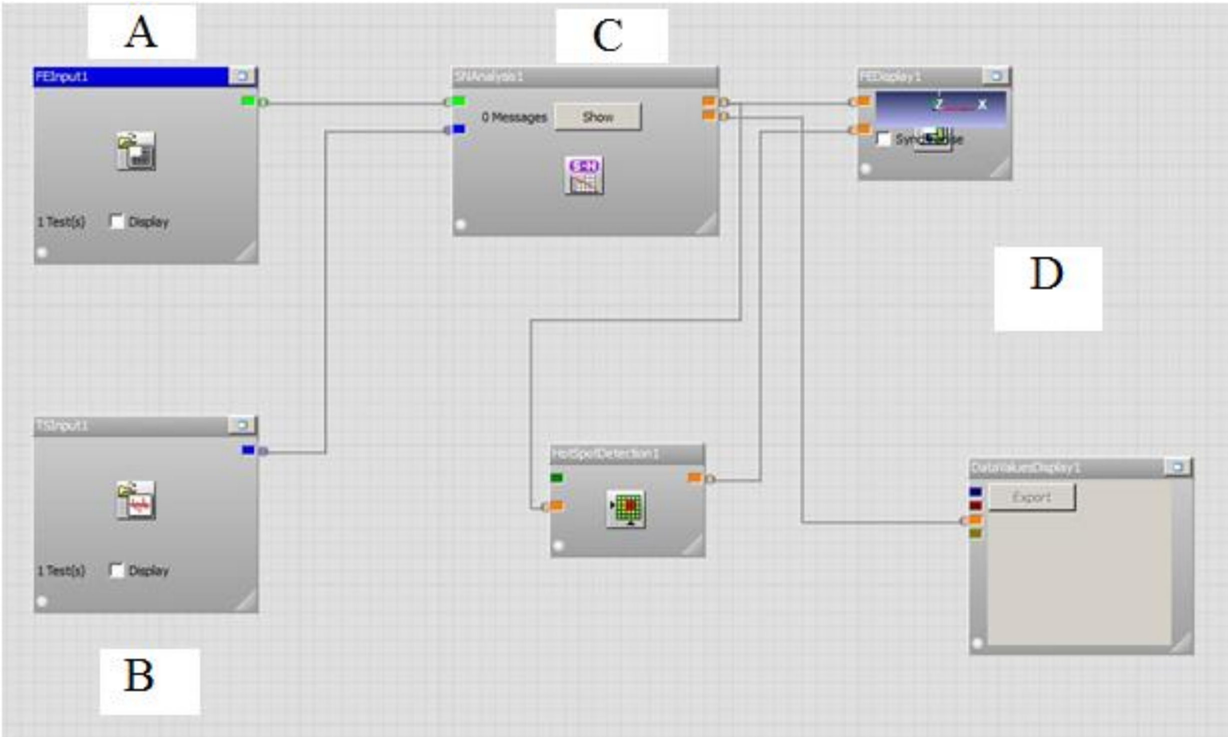

Figure 6: Fatigue Life Process Flow in nCode DesignLife Software

Each load case had a separate FEA outputs, therefore, there was a total of 6 FEA files for 6 total load cases. The 6 FEA files are imported into the 'FEInput' Glyph at location A shown in Figure 6. A durability schedule was created using the dynamic load history data given from the dynamic software. The durability objective is 5000 miles. Various types of courses from proving grounds were used to create the dynamic load time histories. Table 3 shows the breakdown of the duty cycle for the courses simulated. Each course ranges from smooth road to off-road conditions. The dynamic load data was imported in the 'TimeSeries' glyph at location B shown in Figure 6.

Table 3: Duty Cycle from Dynamic Software

Course	Percent of 5000 miles	Required Miles
Road 1	15%	750
Road 2	50%	2500
Road 3	35%	1750

A stress-life (S-N) curve for the three material types/surface treatments (A,B,C) was obtained from the material testing group. The S-N curve was imported into the 'SNAnalysis' Glyph at location C shown in Figure 6. The S-N curves for Design 1, 2, and 3 are shown in Figure 7. Also, the Goodman method was used to calculate the damage from the mean stress. The parameter was selected in the 'SNAnalysis" Glyph as well.

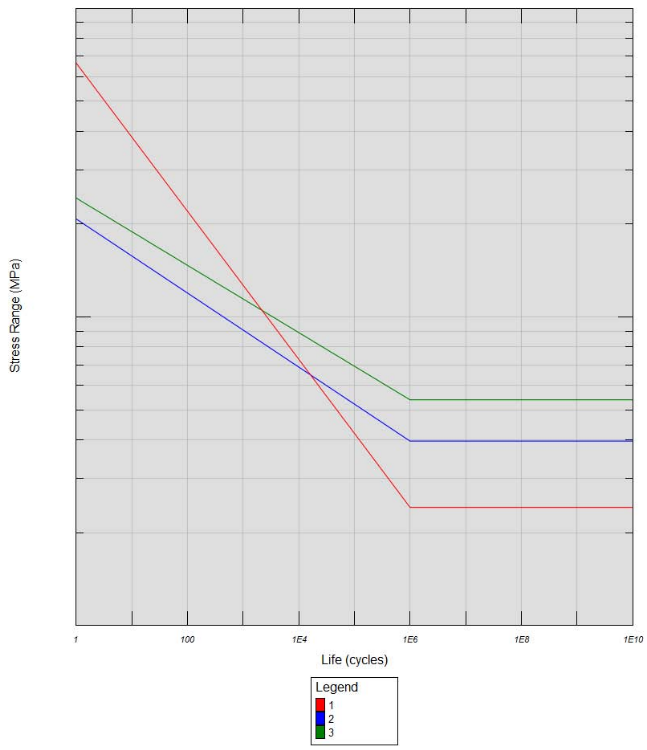

Figure 7: S-N Curves for Design 1, 2, 3

3. Analysis/Results

The three glyphs mentioned from Figure 4 are looped together to output damage results located at location D 'FEResults' glyph. Figures 8 and 9 show the damage contour for the track pin. The close-up figure shows the first location of the initiated crack, highlighted in red. The fracture location matched with the actual fracture location from testing.

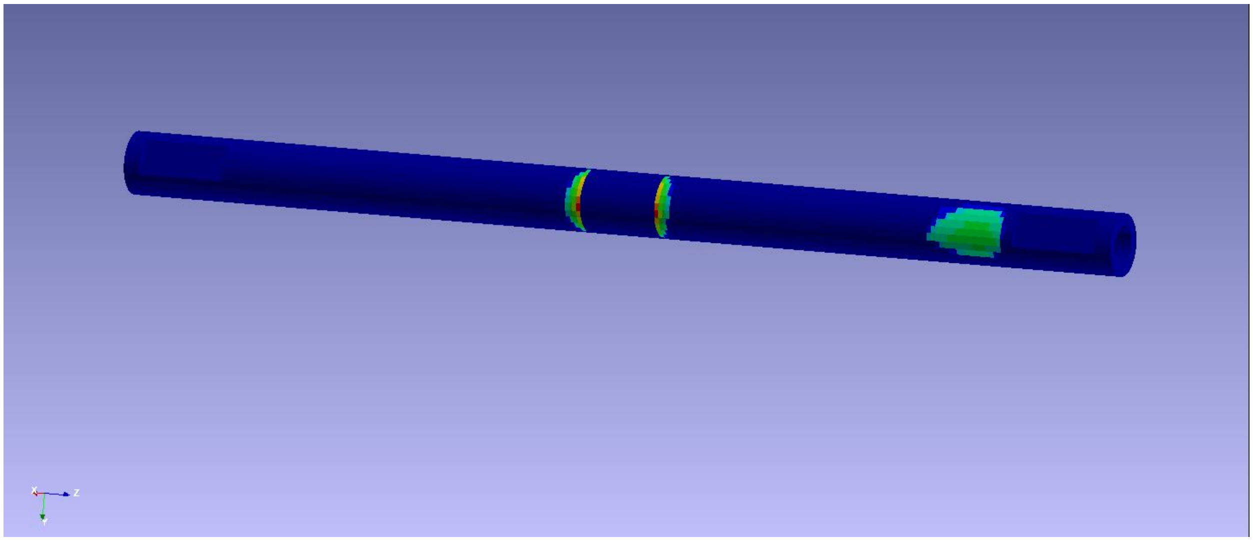

Figure 8: Damage Contour Results for Design 1

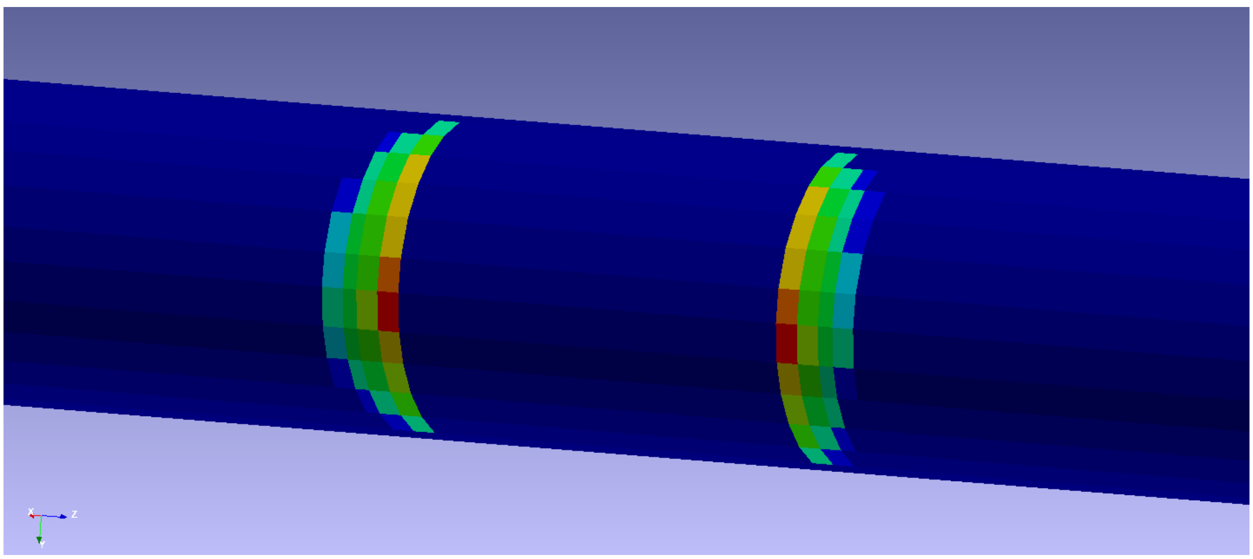

Figure 9: Close-up at the Middle of Track Pin

Ncode predicted a fatigue life value of approximately 1940 miles. This value was close to the average test failures of 1845 miles. The error could be due to several sources including, error in control setup of surface treatments, material flaws, FEA error, dynamic modeling error, S-N curve and material test setup, fatigue life analysis method settings, etc. Considering all those sources of error, a 95 mile difference between the predicted and actual results is acceptable.

Since the predicted value is close to the actual results, an 'A-to-B-to-C' comparison was used to determine the objective of the analysis, whether the other designs meet the durability requirement.

The procedures in Ncode software were done for the other designs and the following results are shown in Table 4.

Table 4: Fatigue Life Predictions

Design	Predicted Fatigue Life (miles)
1	1940
2	27700
3	11800

Both Design 2 and 3 meet the durability requirement. Design 3 was chosen since it has the same thin-wall structure as design 3. Design 2 would be adding mass due to its thick wall structure design. The predicted fatigue life for Design 3 is well over the requirement of 5000 miles, so it is recommended to analyze additional iterations to minimize mass. It is important to note that more physical testing is recommended to further validate the modeling & simulation results.

4. Conclusion

The objective of the project was met by performing a FEA and Fatigue/Fracture analysis to determine if the new designs pass the durability requirement. The original design resulted in a fatigue/fracture life prediction of 1940 miles. The crack initiation failure occurred first at the middle of the track pin. This location and the life prediction were validated close to the actual test data. Both new designs resulted in a prediction greater than the durability objective of 5000 miles. Design 3 would be acceptable to replace the original design, although, additional iterations are recommended to achieve lower mass.

References

1. *A First Course in the Finite Element Method, 4th edition*, by D.L. Logan, Thomson Engineering, 2006.
2. *Altair Hyperworks 10.0 Hypermesh User's Guide: How Element Quality is Calculated.*
3. *Mechanics of Materials, 7th edition*, by James M. Gere and Barry J. Goodna, Cenage Learning, 2009.